

Numerical Simulations for Determining Resistance, Trim, and Sinkage Characteristics of a Tugboat

Utku Cem Karabulut

RESEARCH ARTICLE

Numerical Simulations for Determining Resistance, Trim, and Sinkage Characteristics of a Tugboat

Utku Cem Karabulut 

Department of Naval Architecture and Marine Engineering, Bandirma Onyedi Eylul University, 10200, Balıkesir, Turkey

ABSTRACT

A detailed numerical investigation of the calm-water hydrodynamic performance of an azimuth stern drive (ASD) tugboat is presented. Unsteady Reynolds-Averaged Navier–Stokes (URANS) simulations are performed to predict the total resistance, trim, and sinkage over a wide range of Froude numbers at model scale. The free-surface flow is captured using the Volume of Fluid (VOF) method with a high-resolution interface capturing scheme, while turbulence effects are modeled using the SST $k-\omega$ formulation. To account for dynamic hull response, the vessel is allowed to move freely in heave and pitch through a Dynamic Fluid Body Interaction (DFBI) approach. A systematic mesh and time-step convergence study is conducted to ensure numerical accuracy and solution independence. The numerical predictions of resistance coefficient, trim angle, and sinkage show good agreement with available experimental data, validating the adopted computational framework. In addition to global performance metrics, detailed analyses of the flow field are carried out, including free-surface wave patterns, hull pressure and skin-friction distributions, and stern wake characteristics at the propeller plane. The results provide physical insight into the resistance mechanisms and stern flow features of ASD tugboats operating in calm water. While employing established numerical methods, the primary contribution of this work lies in providing a comprehensively validated dataset and detailed flow physics analysis for a benchmark ASD tugboat—a bluff, unconventional hull form that remains underrepresented in high-fidelity CFD literature. The validated methodology and flow-field findings support the application of CFD as a reliable tool for tugboat hydrodynamic assessment and design optimization.

Keywords: Tugboat hydrodynamics, Calm-water resistance, CFD, URANS, ASD tugboat

How to cite this article: Karabulut, U. C. (2026). Numerical Simulations for Determining Resistance, Trim, and Sinkage Characteristics of a Tugboat. *Seatific*, 6(1), 1–14. <https://doi.org/10.29187/2792-0771.1046>

1. Introduction

Tugboats play a critical role in maritime operations by providing towing, berthing, escorting, and ship-handling services in confined and shallow-water environments (Zhen et al., 2018; Chen et al., 2020). Unlike conventional merchant vessels, tugboats are characterized by relatively short length-to-beam ratios, large displacement-to-length ratios, and the presence of prominent stern appendages such as

skegs and azimuth propulsion units (Allan et al., 2004; Zhang et al., 2022, June). These geometric features, combined with their wide operational speed range and high thrust requirements, give rise to complex hydrodynamic phenomena, including strong viscous effects, pronounced wave-body interactions, and highly three-dimensional stern flow structures (Lutfi et al., 2023, September; Yusfianda et al., 2023, September). Accurate prediction of the calm-water hydrodynamic performance of tugboats therefore

Received 17 December 2025; revised 14 January 2026; accepted 26 January 2026.
Published online 31 January 2026

E-mail address: ukarabulut@bandirma.edu.tr (U. C. Karabulut).

<https://doi.org/10.29187/2792-0771.1046>

2792-0771/© 2026 Published by Yıldız Technical University Press, İstanbul, Türkiye. This is an open access article under the CC BY-NC 4.0 Licence (<https://creativecommons.org/licenses/by-nc/4.0/>).

remains a challenging task, yet it is essential for the efficient design, performance optimization, and operational assessment of modern tug fleets.

While the verification and validation of Computational Fluid Dynamics (CFD) applications for predicting calm-water resistance are well-established for conventional commercial displacement hulls (Wilson et al., 2001; Khan et al., 2025) and high-speed planing crafts (De Luca et al., 2016; Karabulut & Barlas, 2025), and while numerous semi-empirical methodologies exist for these vessel types (Holtrop & Mennen, 1982; Savitsky, 1964), a significant gap remains for unconventional hull forms (Niklas & Prusko, 2019). Tugboats, with their distinct geometric characteristics, deviate markedly from these conventional forms. Consequently, the flow physics governing their resistance, trim, and sinkage are inherently more complex, involving pronounced viscous effects, strong wave-body interactions, and highly three-dimensional stern flows (Lutfi et al., 2023, September). The existing body of validated CFD data and tailored semi-empirical methods for such specialized workboat geometries remains limited. This underscores the necessity for detailed, validated numerical investigations specifically targeting tugboat hydrodynamics to establish reliable CFD frameworks for their design and performance assessment.

The hydrodynamic analysis of tugboats has relied on simulation-based approaches and model-scale experiments conducted in towing tanks. Bernitsas and Kekridis (1985) introduced one of the early simulation-based approaches for analyzing ship towing operations, with particular emphasis on the dynamic stability of the towed vessel. Building on earlier towing simulations, Fitriadhy and Yasukawa (2011) examined the course stability of ship towing systems, providing further insight into the dynamic response of tug-assisted operations. Geerts et al. (2011) investigated interaction forces arising during tug operations, demonstrating the significant influence of hydrodynamic interaction effects in shallow and confined waters.

Early experimental studies provided valuable insight into resistance characteristics, trim and sinkage behavior, and propulsion performance, forming the foundation for empirical design practices. Notably, Brandner (1995) conducted a comprehensive experimental and analytical investigation of omnidirectional stern drive tugboats, highlighting the influence of hull form and appendage configuration on resistance and maneuvering performance. More recently, Zheng et al. (2023) combined experimental measurements with numerical simulations to investigate seakeeping behavior, offering valuable reference

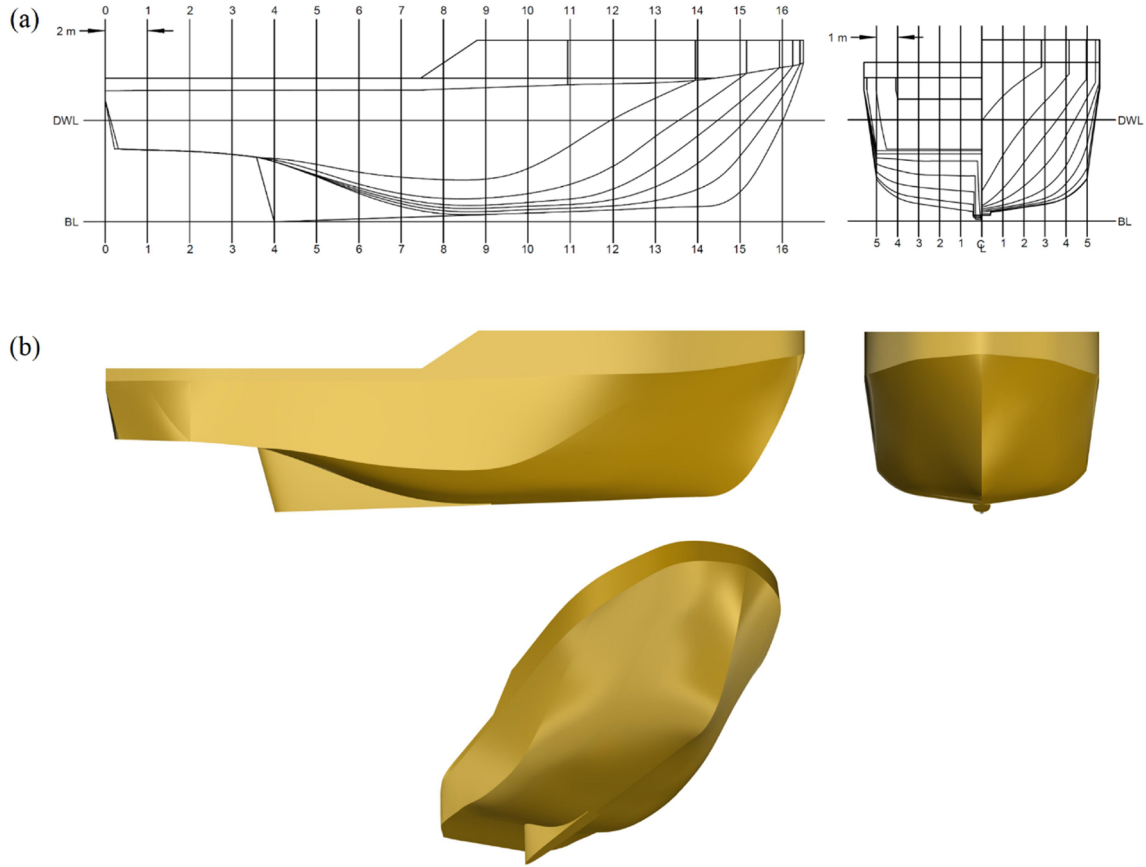
data for the validation of CFD models applied to both wave-induced and calm-water conditions. Such experimental efforts have played a central role in establishing benchmark hull forms and validating theoretical approaches. However, experimental campaigns are inherently time-consuming and costly, and their applicability may be limited when exploring a wide design space or assessing detailed local flow phenomena.

Advances in computational fluid dynamics have significantly enhanced the capability to analyze tugboat hydrodynamics beyond the limitations of empirical and experimental approaches (Aydın et al., 2018; Smoker et al., 2016). Reynolds-Averaged Navier-Stokes (RANS) and Unsteady RANS (URANS) methods, coupled with appropriate turbulence closures and free-surface capturing techniques, have been increasingly applied to workboats and other bluff-hull vessels operating at low to moderate Froude numbers (Weymouth et al., 2005; Carrica et al., 2007). Recent CFD studies on tugboats and related vessel types have demonstrated that reliable prediction of resistance, trim, and sinkage requires careful treatment of mesh resolution, wall modeling, and numerical discretization, as well as consideration of dynamic hull motions (Karabulut et al., 2022; Sarker & Tarafder, 2024). Nevertheless, discrepancies between numerical predictions and experimental data persist, particularly at higher Froude numbers where wave-making and flow separation become increasingly nonlinear, underscoring the need for further validated and systematic CFD investigations (Ozdemir et al., 2021; Hosseini et al., 2024).

In this context, the present study provides a detailed and validated numerical assessment of the calm-water hydrodynamic performance of a benchmark azimuth stern drive tugboat using a well-established URANS-based CFD framework. While the numerical approach (URANS with SST $k-\omega$ turbulence model, VOF free-surface capturing, and DFBI motion modeling) is standard, the novelty and scientific contribution of this work stem from its comprehensive application to this specific, complex hull form. A systematically verified and validated dataset is presented, encompassing global performance metrics (resistance, trim, sinkage) across a wide Froude number range and detailed analyses of the accompanying flow field—including free-surface deformation, hull pressure and shear distributions, and stern wake characteristics. For the ASD tugboat, a geometry characterized by a high displacement-to-length ratio and prominent stern appendages, such a comprehensive high-fidelity dataset has been lacking. This work, therefore, addresses a gap in the literature by establishing a reliable CFD benchmark and providing new

Table 1. Main particulars of the tugboat.

Hull Particulars	Symbol	Unit	Full Scale	Model Scale
Scale Factor	λ	-	1	9.78
Length Overall	L_{OA}	m	33	3.374
Length of Waterline	L_{WL}	m	32	3.272
Length Between Perpendiculars	L_{BP}	m	27.5	2.812
Draught	T	m	4.80	0.491
Beam of Waterline	B	m	11.2	1.145
Displacement	Δ	kg	8.16×10^5	873
Wetted Area	S	m ²	469	4.9
Longitudinal Center of Buoyancy (from stern)	LCB	m	16.14	1.65

**Fig. 1.** Geometry of the tugboat. (a) Lines plan, (b) CAD plan.

physical insights into the hydrodynamic behavior of these specialized vessels, thereby supporting future design and analysis efforts.

2. Methodology

2.1. Geometry and simulation cases

The vessel under investigation is a conventional azimuth stern drive (ASD) tug, designed for operations requiring high maneuverability and bollard pull capacity. The hull form design was initially published by Brandner (1995), and scaled model experiments were subsequently conducted by Zheng et al. (2023).

The main particulars of the full-scale and model-scale tugboat are presented in Table 1. A three-dimensional (3D) model of the tugboat was created using its two-dimensional (2D) lines plan. The geometry of the model is presented in Fig. 1.

Numerical simulations with a model-scale tugboat were performed at seven different flow speeds. Table 2 summarizes all cases. Here, V represents the velocity of the tugboat and Reynolds number (Re) and Froude Number (Fr) calculated from the following equations:

$$Re = \frac{\rho V L_{WL}}{\mu} \quad (1)$$

Table 2. Numerical simulation cases.

V (m/s)	Re (Million)	Fr
0.600	1.73	0.106
0.98	2.82	0.173
1.309	3.76	0.231
1.473	4.23	0.260
1.632	4.69	0.288
1.881	5.40	0.332
2.125	6.10	0.375

$$Fr = \frac{V}{\sqrt{gL_{WL}}} \quad (2)$$

Here ρ and μ are the density and dynamic viscosity of water, respectively, while g is the acceleration due to gravity.

2.2. Mathematical model

The viscous flow around the tugboat hull was modeled by solving the three-dimensional, incompressible, Unsteady Reynolds-Averaged Navier–Stokes (URANS) equations. The continuity and momentum equations are given in Cartesian tensor form as:

$$\frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0 \quad (3)$$

$$\begin{aligned} \frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{u}_i \bar{u}_j) = & -\frac{\partial \bar{p}}{\partial x_j} \\ & + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \right] \end{aligned} \quad (4)$$

where $\bar{u}_i$ and $\bar{p}$ are the mean velocity and pressure components, ρ is the fluid density, μ is the dynamic viscosity, and $\rho \overline{u'_i u'_j}$ represents the Reynolds stress tensor (Ferziger et al., 2019).

The Reynolds stress terms were closed using the two-equation Shear Stress Transport (SST) k - ω turbulence model (Menter, 1994). The model solves transport equations for the turbulent kinetic energy k and the specific dissipation rate ω :

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = p_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right] \quad (5)$$

$$\begin{aligned} \frac{\partial \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} = & \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] \\ & + 2(1 - F_1) \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_i} \end{aligned} \quad (6)$$

where p_k is the production rate of turbulent kinetic energy, ν_T is the turbulent eddy viscosity, and F_1 is

a blending function that activates the standard k - ω model near the wall and transitions to the k - ϵ model in the free stream. The constants α , β , β^* , σ_k , σ_ω are model coefficients (Menter, 1994). This formulation is particularly suited for capturing complex flow separation and adverse pressure gradients present in the stern and skeg regions of the tugboat (ITTC, 2011).

The interface between the water and air was captured using the Volume of Fluid (VOF) method (Hirt & Nichols, 1981). The method introduces a scalar volume fraction, α , defined as the fraction of the cell volume occupied by the primary phase (water). The transport equation for α is:

$$\frac{\partial \alpha_i}{\partial t} + u_j \frac{\partial \alpha_i}{\partial x_j} = 0 \quad (7)$$

The volume fraction transport equation was solved using the High-Resolution Interface Capturing (HRIC) scheme (Muzaferija, 1998) to maintain a sharp interface while mitigating numerical diffusion. The fluid properties in each cell are calculated as a weighted average based on the volume fraction:

$$\rho = \alpha \rho_{water} + (1 - \alpha) \rho_{air} \quad (8)$$

$$\mu = \alpha \mu_{water} + (1 - \alpha) \mu_{air} \quad (9)$$

To predict the vessel's dynamic response, the hull was allowed to move freely in heave and pitch (trim) using a Dynamic Fluid Body Interaction (DFBI) model (CD-ADAPCO, 2017). Pressure-velocity coupling was achieved using the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm of Patankar and Spalding (1983). The governing equations were discretized using a cell-centered finite volume method (FVM). Second-order upwind schemes were employed for the convection terms of momentum, turbulence, and volume fraction equations, while a second-order implicit scheme was used for temporal discretization. All numerical simulations were performed using the commercial CFD software Siemens Star-CCM+.

2.3. Computational domain

The computational domain was designed to accurately capture the far-field flow while ensuring minimal influence from the boundaries on the near-hull solution. The computational domain was designed to accurately capture the far-field flow while ensuring minimal influence from the boundaries on the near-hull solution. The extents of the domain were selected following established guidelines for marine CFD applications, particularly those recommended by the International Towing Tank Conference (ITTC,

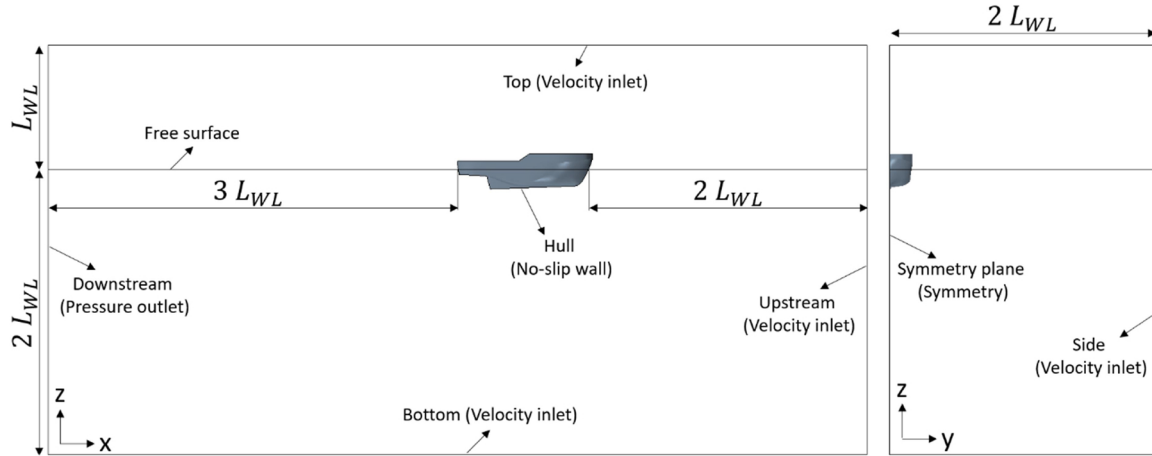


Fig. 2. Domain and boundary conditions.

2011). These guidelines aim to minimize numerical wave reflection and blockage effects, ensuring that boundary conditions do not artificially influence key results such as resistance, trim, sinkage, and wave patterns. The chosen dimensions are consistent with common practice for simulations of displacement hulls at model scale (e.g., Wilson et al., 2001; Carica et al., 2007). As shown in Fig. 2, the domain extends $2 \times L_{WL}$ upstream of the bow and $3 \times L_{WL}$ downstream of the stern. Side boundary is located $2 \times L_{WL}$ from the centerplane, while the top and bottom boundaries are positioned $1 \times L_{WL}$ above and $2 \times L_{WL}$ below from the still waterplane, respectively. The boundary conditions were assigned as follows: a free-stream velocity condition was applied at the inlet (upstream), side, top, and bottom boundaries; a pressure outlet condition was specified at the downstream boundary; and a no-slip wall condition was enforced on the hull surface. The symmetry plane was defined along the vessel's centerline.

The volume mesh was generated using the unstructured, hexahedral-dominant trimmed-cell mesher within Siemens Star-CCM+. The background cell size was set to $0.32 \times L_{WL}$. A systematic, multi-level refinement strategy was then implemented to resolve the critical flow features. The hull surface was discretized with a target cell size of $0.005 \times L_{WL}$ to adequately capture its curvature. Given the hydrodynamic importance of the stern appendages, the skeg surface received an additional local refinement, reducing the target size to $0.00125 \times L_{WL}$ to accurately resolve its sharp geometry and the associated complex flow separation. To sharply capture the air-water interface, a box-shaped control volume encompassing the expected free-surface deformation was defined, extending from the upstream to the downstream and laterally to the domain sides. Within this region, the

mesh was refined in the vertical direction to a target cell size of $0.00125 \times L_{WL}$. To capture the wake field, a triangular volumetric refinement region was created. This region originates at the bow, extends $1.2 \times L_{WL}$ downstream from the stern, and employs a target cell size of $0.01 \times L_{WL}$.

For boundary layer resolution, a prism layer mesh consisting of seven layers with a geometric stretch factor of 1.2 was applied on the wetted hull surface. Crucially, for each simulation, the total thickness of this prism layer stack was adjusted to achieve an average target wall unit of $y^+ \approx 50$ at the first cell centroid across the hull. This value is within the recommended range (30–100) for the use of wall functions with the “All y^+ wall treatment” in Star-CCM+. This approach represents a computationally efficient and validated methodology for predicting integral forces and moments on ship hulls (ITTC, 2011), providing a practical compromise that avoids the prohibitive cost of maintaining a $y^+ < 5$ across the entire hull under dynamic motion. The final mesh for the high-fidelity case contained approximately 3.0 million cells, which is shown in Fig. 3. The resulting wall y^+ distributions for the extreme Froude numbers, confirming the attainment of the target, are presented in Fig. 4.

2.4. Computational details

All numerical simulations were performed using the commercial CFD software Siemens Star-CCM+ (version 2022.1). The computations were conducted on a personal computer equipped with a 6-core processor and 32 GB of RAM. The simulations were run in parallel, utilizing all available computational cores.

For the production cases (fine mesh, $\Delta t = 0.010$ s), each simulation required approximately 96 hours of

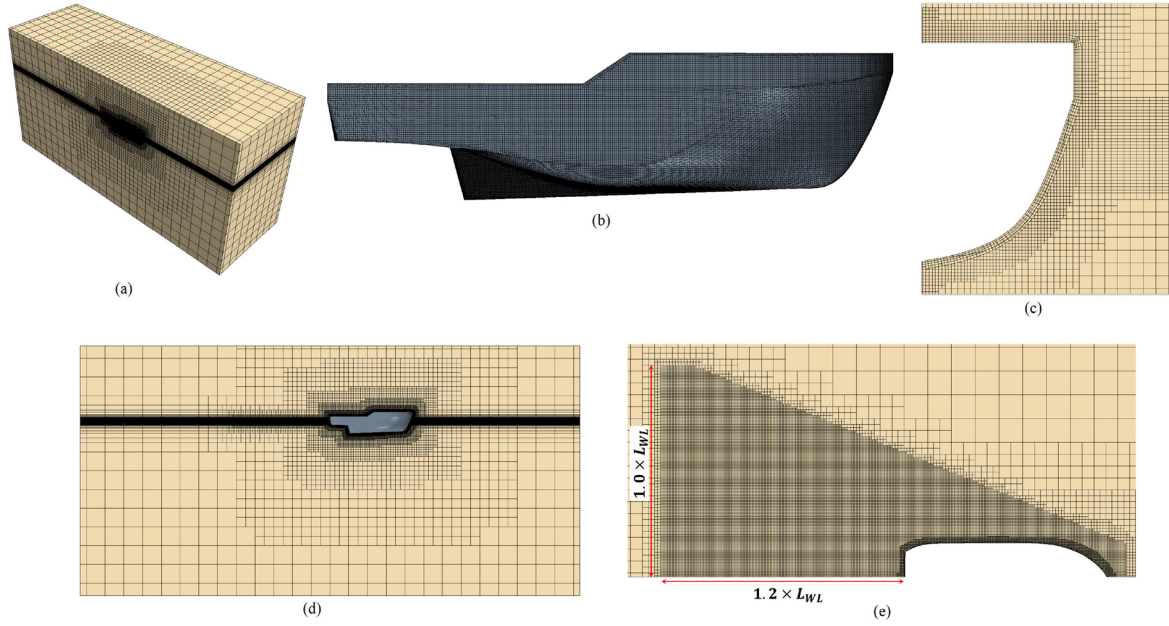


Fig. 3. Volume mesh: (a) Perspective view, (b) Mesh on the hull surface, (c) yz-plane cut at amidships, (d) xz-plane cut at center plane, (e). xy-plane cut on the still water plane.

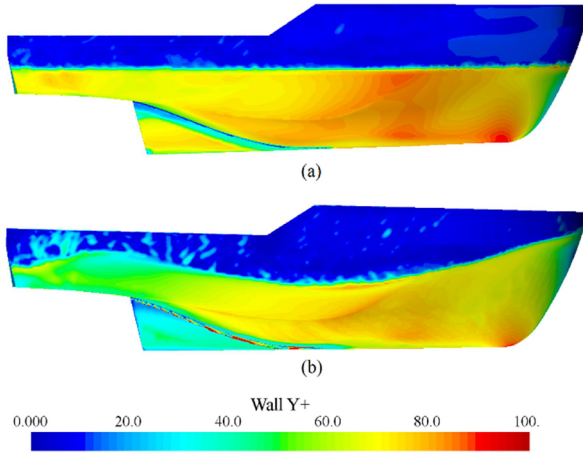


Fig. 4. Wall y^+ distribution on the hull: (a) $Fr = 0.106$, (b) $Fr = 0.375$.

wall-clock time to achieve a fully developed, statistically steady solution for resistance, trim, and sinkage. Fig. 5 shows convergence history of C_T at $Fr = 0.375$.

3. Verification

3.1. Verification methodology

The verification study was conducted to estimate the numerical uncertainties stemming from spatial (grid) and temporal discretization. The procedure adhered to established best practices in computational fluid dynamics for ship hydrodynamics, including the guidelines of the International Towing Tank Con-

ference (ITTC, 2024) and the uncertainty analysis methodology of Xing and Stern (2010). The study was performed at a single, representative condition of $Fr = 0.288$, which corresponds to a typical full-scale operational speed ($V \approx 10$ knots) and features a balanced mix of flow physics (Molyneux, 2006).

The study is based on obtaining multiple solutions using systematically refined grid sizes and time steps. For the grid convergence study, three grids were generated: a coarse grid, a medium grid, and a fine grid. The characteristic grid spacing for these grids are Δx_3 , Δx_2 and Δx_1 , respectively, where $\Delta x_1 < \Delta x_2 < \Delta x_3$.

Δ . The grid refinement factor is defined as the ratio of the grid spacings, $r_{21} = \Delta x_2 / \Delta x_1$ and $r_{32} = \Delta x_3 / \Delta x_2$. An identical systematic refinement approach was applied for the time-step convergence study, with three time-step sizes Δt_3 , Δt_2 and Δt_1 .

For a generic solution variable ϕ (e.g., resistance coefficient, trim angle, sinkage), the solutions on the three grids are ϕ_1 (fine), ϕ_2 (medium), and ϕ_3 (coarse). The convergence condition is assessed via the convergence ratio R_G :

$$R_G = \frac{\phi_2 - \phi_1}{\phi_3 - \phi_2} \quad (10)$$

The convergence is classified as:

- Monotonic Convergence: $0 < R_G < 1$
- Oscillatory Convergence: $R_G < 0$
- Divergence: $1 < R_G$

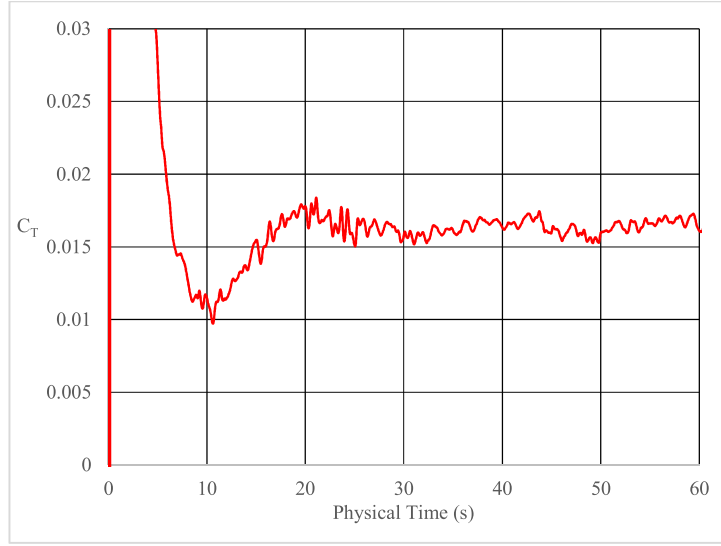


Fig. 5. Convergence history of resistance coefficient C_T at $Fr = 0.375$.

For cases exhibiting monotonic convergence, the generalized Richardson extrapolation method is used to estimate the observed order of convergence, p_a , extrapolated value, ϕ_{ext}^{21} , factor of safety, FS , and numerical uncertainty, U_G , based on [ITTC \(2024\)](#). In this method, the apparent order of accuracy, p_a , can be calculated iteratively from the following equations ([Celik et al., 2008](#)):

$$p_a = \frac{1}{\ln r_{21}} \ln \left| \frac{\varepsilon_{32}}{\varepsilon_{21}} \right| + q(p_a) \quad (11)$$

$$q(p_a) = \ln \left(\frac{r_{21}^{p_a} - s}{r_{32}^{p_a} - s} \right) \quad (12)$$

$$s = \text{sgn} \left(\frac{\varepsilon_{32}}{\varepsilon_{21}} \right) \quad (13)$$

where r_{21} and r_{32} are the refinement ratios of consecutive meshes or time steps. ε_{21} and ε_{32} are defined as:

$$\varepsilon_{21} = \phi_1 - \phi_2 \quad (14)$$

$$\varepsilon_{32} = \phi_2 - \phi_3 \quad (15)$$

The extrapolated value, ϕ_{ext}^{21} , is determined from:

$$\phi_{ext}^{21} = \frac{r_{21}^{p_a} \phi_1 - \phi_2}{r_{21}^{p_a} - 1} \quad (16)$$

Finally, the approximate relative error, the extrapolated relative error, and the GCI of the fine grid are calculated from ([Celik et al., 2008](#)):

$$e_a^{21} = \left| \frac{\phi_1 - \phi_2}{\phi_2} \right| \quad (17)$$

$$e_{ext}^{21} = \left| \frac{\phi_{ext}^{21} - \phi_1}{\phi_{ext}^{21}} \right| \quad (18)$$

$$GCI_{fine}^{21} = \frac{FS e_a^{21}}{r_{21}^{p_a} - 1} \quad (19)$$

Where $U_G = GCI_{fine}^{21}$, and FS is the factor of safety, which is calculated based on [Xing and Stern \(2010\)](#):

$$FS = \begin{cases} 2.45 - 0.85P & \rightarrow 0 < P \leq 1 \\ 16.4P - 14.8 & \rightarrow 1 < P \end{cases} \quad (20)$$

Here, P is the ratio of the theoretical order of accuracy to the apparent order of accuracy. When oscillatory convergence is achieved, numerical uncertainty, U_G , is determined based on [Stern et al. \(2001\)](#):

$$U_G = \frac{1}{2} \max(|\phi_2 - \phi_1|, |\phi_3 - \phi_2|, |\phi_3 - \phi_1|) \quad (21)$$

3.2. Mesh and time-step convergence results

The total resistance coefficient, C_T a primary parameter of interest, is defined as:

$$\frac{R_T}{0.5\rho SV^2} \quad (22)$$

where R_T is the total resistance, and S is the wetted surface area of the hull, as given in [Table 1](#).

First, mesh convergence was evaluated using three systematically refined grids: coarse (721,782 cells), medium (1,485,666 cells), and fine (3,068,731 cells). [Table 3](#) shows the results for grid convergence analysis. According to the results, monotonic convergence

Table 3. Grid convergence results at $Fr = 0.288$.

Simulation	Cell Count	C_T	Trim ($^\circ$)	Sinkage (mm)
Coarse	721,782	0.1027	0.739	21.4
Medium	1,485,666	0.1043	0.757	20.1
Fine	3,068,731	0.1054	0.769	21.0
Convergence condition		Monotonic	Monotonic	Oscillatory
p_a		1.574	1.701	N/A
FS		1.781	1.727	N/A
U_G (%)		4.0	5.3	3.1

Table 4. Time-step convergence results at $Fr = 0.288$.

Simulation	Δt (s)	C_T	Trim ($^\circ$)	Sinkage (mm)
Coarse	0.020	0.1051	0.774	21.1
Medium	0.015	0.1056	0.771	21.0
Fine	0.010	0.1054	0.769	21.0
Convergence condition		Oscillatory	Monotonic	Oscillatory
p_a		N/A	2.21	N/A
FS		N/A	3.301	N/A
U_T (%)		0.2	0.6	0.01

is achieved for C_T and trim, while oscillatory convergence was achieved for sinkage. The uncertainty related to grid size in C_T , trim and Sinkage are calculated as 4.0%, 5.3% and 3.1%, respectively. Consequently, the fine mesh was selected for all subsequent simulations to ensure the highest practical spatial accuracy.

Subsequently, time-step convergence was investigated to ensure temporal independence. The selection of time-step sizes was guided by the ITTC (2011) recommended practice, which suggests a non-dimensional time step (Δt^*) in the range of 0.005 to 0.01, defined as $\Delta t^* = \Delta t(U/L)$, where U is the flow velocity and L is the characteristic length (taken as L_{WL}). For the convergence study case at $Fr = 0.288$ ($V = 1.632$ m/s and $L_{WL} = 3.272$ m), this corresponds to a dimensional time-step range of approximately 0.010 s to 0.020 s. Accordingly, simulations were performed with three different time-step sizes within this range: $\Delta t = 0.020$ s, 0.015 s, and 0.010 s. Table 4 shows the results for time-step convergence analysis, where U_T is the numerical uncertainty due to the time-step. The results show that the uncertainty related to time step in C_T , trim and Sinkage are below 1%. Based on this analysis, a time step of $\Delta t = 0.010$ s was selected for all subsequent cases.

4. Results and discussion

4.1. Resistance, trim and sinkage

The calm-water hydrodynamic performance of the tugboat, characterized by its total resistance coefficient C_T , trim angle, and sinkage at the center of gravity, was numerically investigated across the

Froude number range of 0.106 to 0.375. The results from the fine-mesh, fine-time-step simulations are compared with the experimental data published by Zheng et al. (2023) in Fig. 6.

Fig. 6(a) presents the variation of the total resistance coefficient C_T with Froude number. The numerical predictions closely follow the experimental trend, successfully capturing the characteristic resistance curve of a conventional displacement hull. The agreement is quantified by a mean absolute percentage error (MAPE) of 2.9% across the speed range, with the largest individual discrepancies of 5.8% occurring at the lowest speed ($Fr = 0.106$), and 6.9% occurring at the highest speed ($Fr = 0.375$).

The comparison of trim angle is shown in Fig. 6(b). The numerical model reproduces the trend measured experimentally, with the trim angle increasing with Froude number as expected. The computational results show consistent alignment with the physical measurements, with a MAPE of 6.0% for trim. The maximum error was 9.3% at the highest speed ($Fr = 0.375$), where wave-making effects and dynamic motions are most pronounced.

Sinkage at the center of gravity is presented in Fig. 6(c). The sinkage increases with Froude number due to dynamic pressure changes along the hull. The computational fluid dynamics (CFD) results show good agreement with the experimental measurements, with a MAPE of 7.2% for sinkage. It should be noted that the relative error at the lowest speed ($Fr = 0.106$) is 27.3%; however, this corresponds to an absolute discrepancy of less than 0.5 mm in model scale, reflecting the challenge of accurately measuring and predicting very small sinkage values near the static condition. Excluding this lowest speed point, the MAPE for sinkage reduces to 3.8%.

It is noted that the comparison error for trim and sinkage does not vary monotonically with Froude number but shows fluctuation. This behavior is physically consistent and reflects the changing dominance of underlying hydrodynamic mechanisms. At lower speeds ($Fr < 0.2$), performance is governed by viscous pressure fields and boundary layer development, which are well-captured by the RANS model. At higher speeds, dynamic wave-making pressures—characterized by nonlinear interactions between the bluff bow, skeg, and stern wave systems—become the primary driver for trim and sinkage. The tugboat's specific geometry, with its high displacement-to-length ratio and prominent stern appendages, makes this transition particularly sensitive, leading to local fluctuations in error as the flow regime shifts.

Overall, the numerical simulations demonstrate a consistent and reliable ability to predict the key calm-water performance metrics of the tugboat. The

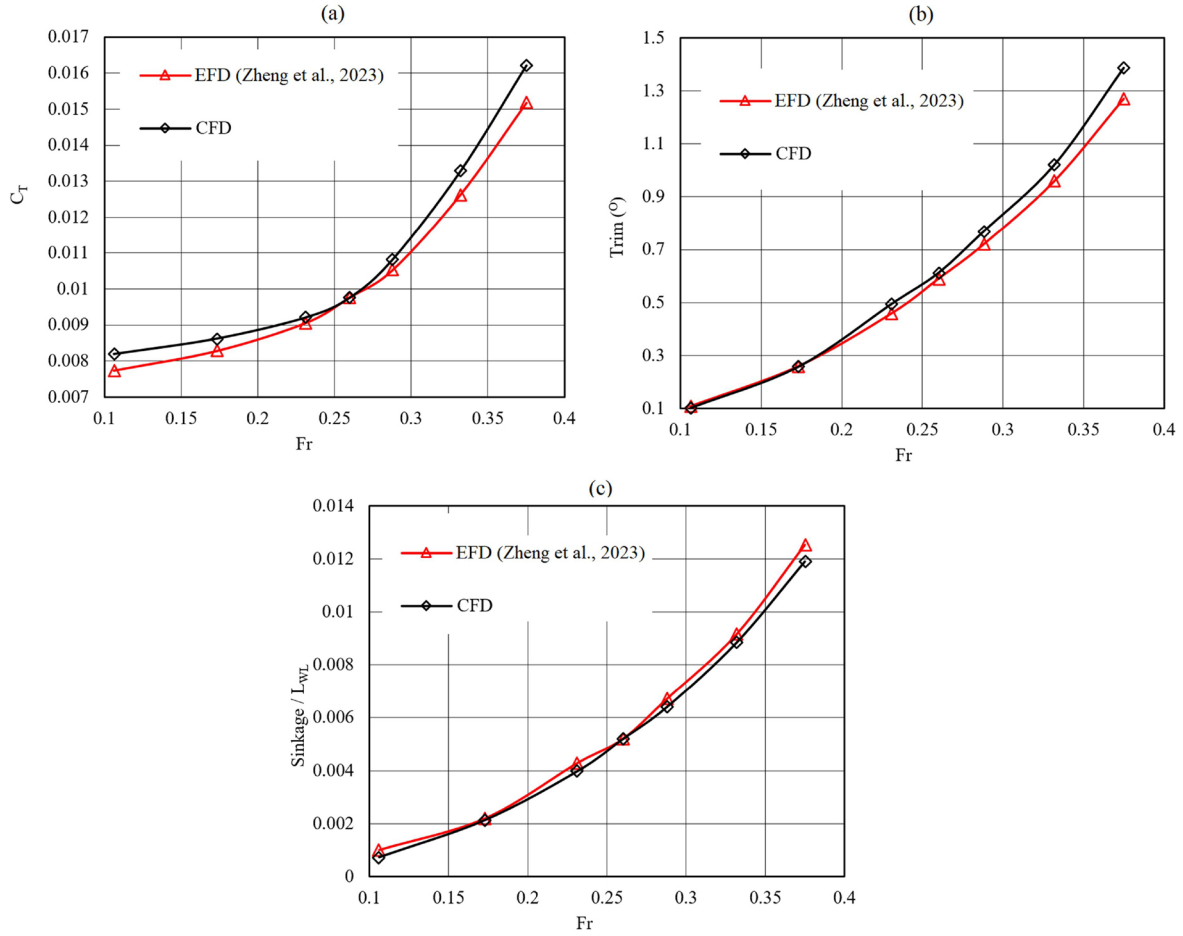


Fig. 6. Comparison of the experimental (EFD) and numerical (CFD) results for (a) resistance coefficient, (b) trim, and (c) sinkage. Experimental results are from Zheng et al., 2023.

agreement with the experimental data from Zheng et al. (2023) validates the selected numerical methodology, including the URANS approach with the SST $k-\omega$ turbulence model, the VOF free-surface treatment, and the DFBI motion model, as a practical tool for the hydrodynamic assessment of this type of vessel.

4.2. Flow field around tugboat

A detailed analysis of the flow field around the tugboat hull was conducted to elucidate the hydrodynamic mechanisms governing resistance, trim, and sinkage across the investigated Froude number range. Three key characteristics were the main parameters of interest: the evolution of the free-surface wave pattern, the hull pressure and skin-friction distributions, and the wake structure at the propeller plane. Together, these elements are critical for a comprehensive understanding of ASD tugboat performance in calm water.

Fig. 7 presents the computed free-surface elevations around the hull for selected Froude numbers. At the lowest speed condition ($Fr = 0.106$), the wave system is relatively weak and symmetric, consisting of a modest divergent bow wave and a gently recovering stern wave. The free surface remains largely undisturbed along the midship region, indicating that viscous resistance dominates the total resistance at this speed. As the Froude number increases, wave-making effects become increasingly significant. Distinct bow and shoulder wave systems develop, accompanied by a pronounced wave trough near the bow and an amplified stern wave depression. At higher speeds ($Fr \geq 0.288$), strong divergent and transverse wave components are clearly observed, with increased wave steepness and downstream propagation. These features are characteristic of displacement-type hulls operating at moderate to high Froude numbers and directly correspond to the observed rise in the total resistance coefficient reported in Section 4.1.

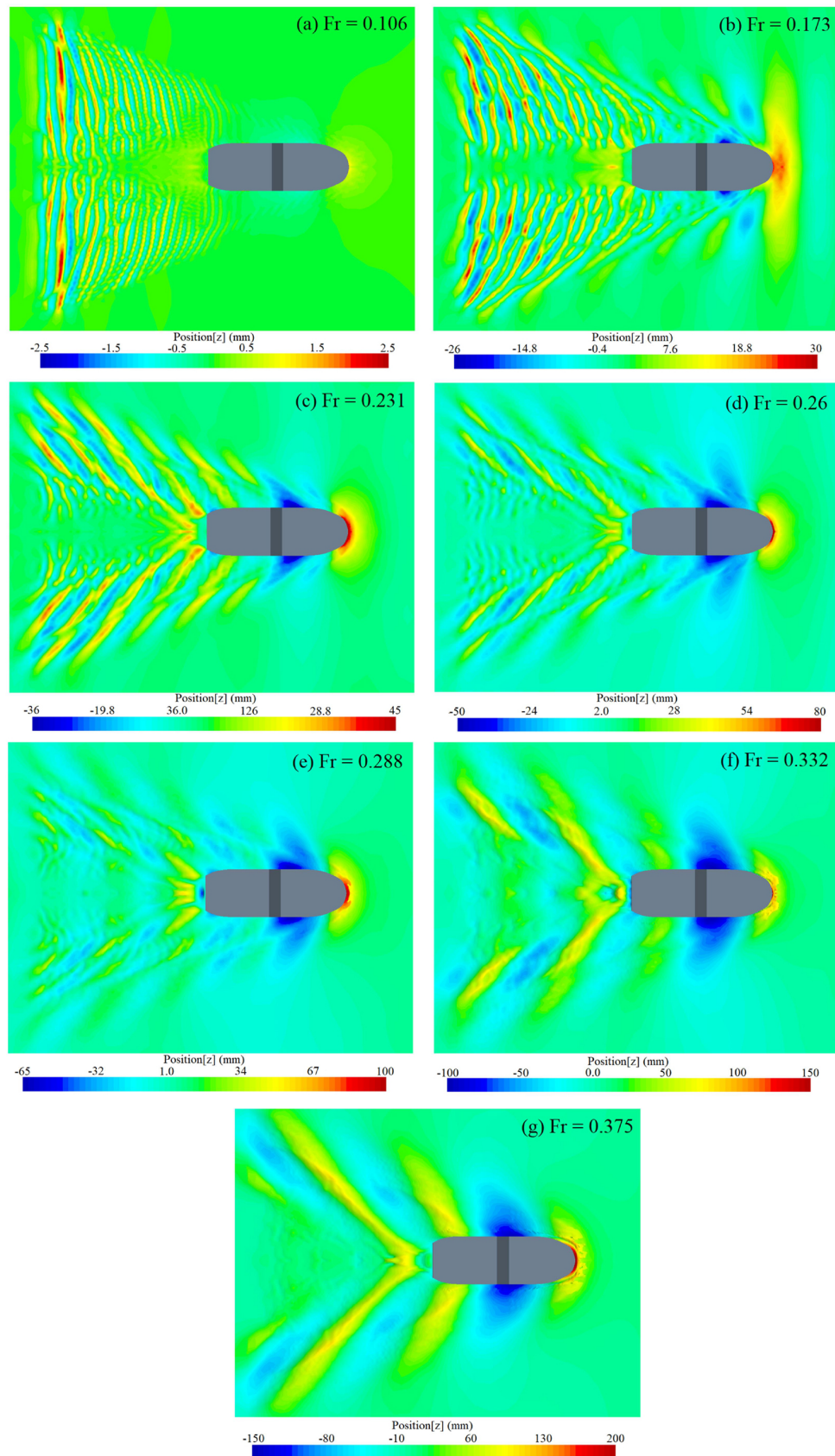


Fig. 7. Free surface deformations around the tugboat at various Froude numbers.

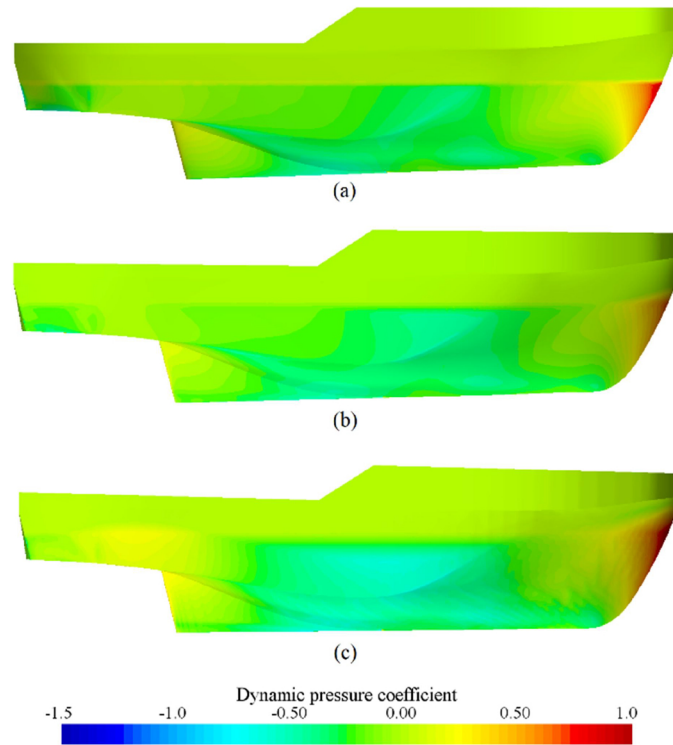


Fig. 8. Dynamic pressure distribution on the tugboat: (a) $Fr = 0.106$, (b) $Fr = 0.26$, (c) $Fr = 0.375$.

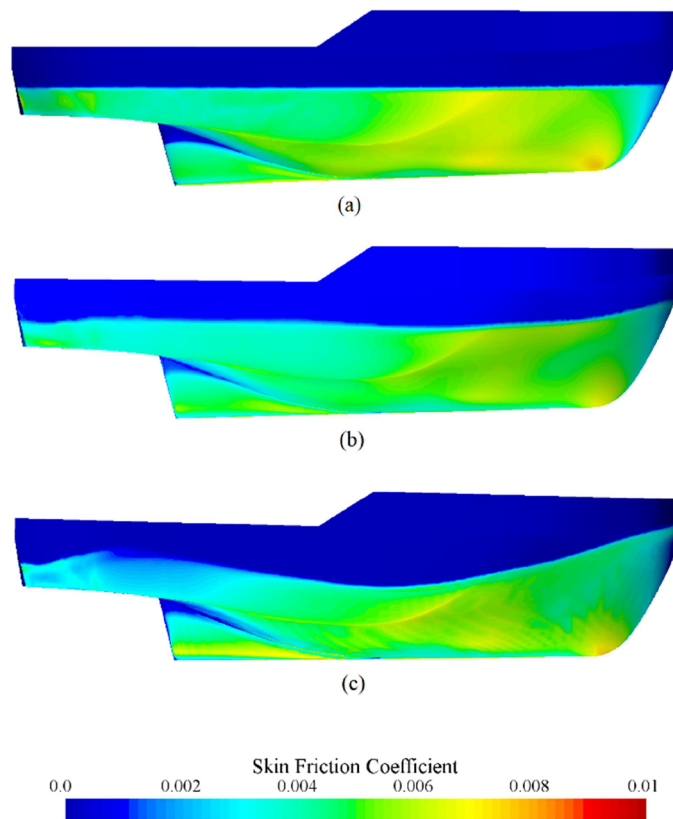


Fig. 9. Skin friction distribution on the tugboat: (a) $Fr = 0.106$, (b) $Fr = 0.26$, (c) $Fr = 0.375$.

The dynamic pressure distributions on the hull surface for representative speeds are illustrated in Fig. 8. At low Froude number, the pressure field is largely governed by hydrostatic effects, with elevated pressures concentrated around the bow stagnation region and a relatively smooth pressure gradient along the hull. As speed increases, the influence of dynamic pressure becomes more pronounced. High-pressure regions intensify at the forebody due to increased stagnation effects, while extended low-pressure zones develop along the aft shoulder and stern regions. Notably, the skeg and stern appendage areas experience localized pressure peaks, particularly at $Fr = 0.26$ and $Fr = 0.375$, reflecting strong flow acceleration and deceleration in these regions. These pressure asymmetries contribute significantly to the longitudinal force balance and play an important role in the predicted trim behavior of the tugboat.

Fig. 9 shows the distribution of skin-friction coefficient on the hull surface. At lower speeds, the skin-friction distribution is relatively uniform, with higher values concentrated near the bow and forward bottom region due to thinner boundary layers and higher local velocities. With increasing Froude number, the magnitude of skin friction increases over the entire wetted surface, especially along the flat bottom and stern regions where flow acceleration and turbulence intensity are enhanced. Elevated skin-friction levels are also observed near the skeg and stern areas, indicating strong viscous effects associated with flow curvature and possible local separation.

The wake structure downstream of the hull is examined through normalized streamwise velocity contours at the propeller plane ($x/L_{WL} = 0.11$, where x is measured from stern), as shown in Fig. 10. At $Fr = 0.106$, the wake is relatively narrow, with a modest velocity deficit centered along the hull centerline. As the speed increases, the wake expands both vertically and laterally, and the velocity deficit becomes more pronounced. At higher Froude numbers, complex wake patterns emerge, characterized by strong velocity gradients, low-speed regions behind the skeg, and accelerated flow zones near the outer portions of the propeller disk. These wake features are of particular relevance for ASD tugboats, as they directly influence inflow conditions of the propulsors, propulsion efficiency, and maneuvering performance.

Overall, the flow field analysis demonstrates that the numerical model successfully captures the essential hydrodynamic features associated with tugboat operation across a wide speed range. The predicted free-surface patterns, pressure and shear distributions, and wake characteristics are physically consistent and provide valuable insight into the resistance mechanisms and dynamic behavior of the vessel.

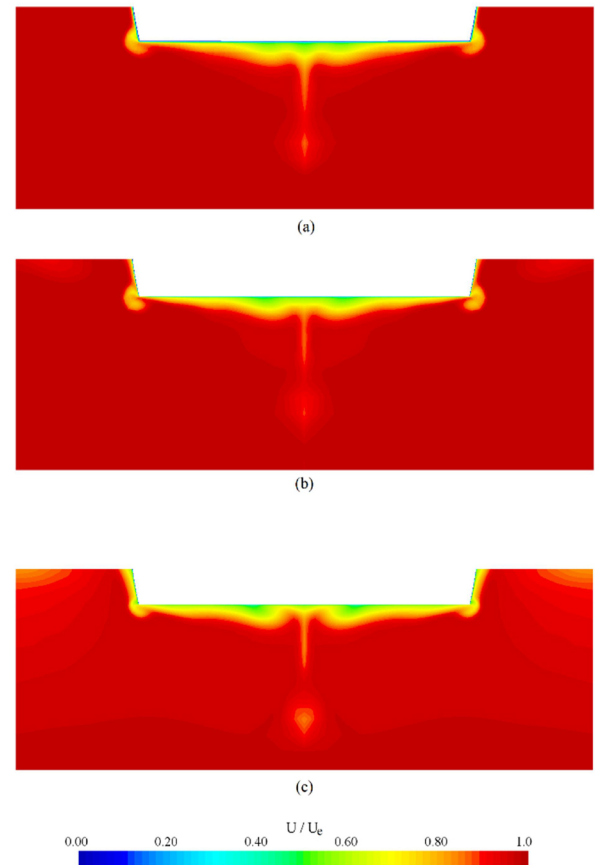


Fig. 10. Normalized streamwise velocity distributions at propeller plane: (a) $Fr = 0.106$, (b) $Fr = 0.26$, (c) $Fr = 0.375$.

This detailed flow-field information complements the global performance metrics presented earlier and further confirms the robustness of the adopted URANS–VOF–DFBI modeling framework for tugboat hydrodynamic analysis.

5. Conclusion

This study presented a detailed numerical investigation into the calm-water hydrodynamic performance of an Azimuth Stern Drive (ASD) tugboat using an unsteady RANS-based CFD approach. The methodology was rigorously verified and validated against experimental data, providing both global performance metrics and detailed insights into the accompanying flow physics.

The key findings of this work are summarized as follows:

- A systematic verification study confirmed the spatial and temporal convergence of the numerical solutions. The estimated numerical uncertainties for the fine-grid solutions were found to be within

3–5% for resistance, trim and sinkage establishing confidence in the predictions.

- The validated simulations demonstrated good agreement with experimental measurements across a range of Froude numbers. The total resistance coefficient, trim, and sinkage were predicted with mean absolute percentage errors (MAPE) of 2.9%, 6.0%, and 3.8%, respectively, confirming the accuracy of the adopted URANS-VOF-DFBI framework for this hull type.
- Detailed flow-field analysis revealed the evolution of resistance components with speed. At lower Froude numbers, resistance was dominated by viscous effects, while wave-making became increasingly significant at higher speeds. The analysis of pressure and skin-friction distributions highlighted the critical role of the bluff bow and stern appendages in governing the longitudinal force balance and dynamic trim.
- The stern wake analysis at the propeller plane documented a complex, three-dimensional flow structure with significant velocity deficits. This characterization provides a valuable baseline for understanding propulsor inflow conditions in ASD tug configurations.

This study is conducted entirely at model scale, which presents certain limitations for direct industrial application. Specifically, it does not address scale effects, form factor decomposition, correlation allowances, or full-scale resistance prediction—key elements required for translating model-scale results to actual vessel performance. To bridge this gap and extend the practical utility of the findings, future research should focus on several targeted areas. First, a dedicated investigation into scale effects for tugboat hulls using CFD at full scale is needed to quantify differences in resistance components and flow characteristics. Subsequently, the validated CFD model should be integrated with propeller simulations to study hull-propulsor interaction and predict full-scale performance metrics, such as delivered power and bollard pull capacity. Finally, extending the methodology to simulate dynamic maneuvering and operations in waves would better represent the true operational profile of tugboats, providing a more comprehensive tool for design and assessment.

In summary, this work establishes a robust and validated CFD methodology for the hydrodynamic analysis of ASD tugboats in calm water. The comprehensive dataset and physical insights provided form a solid foundation for subsequent design optimization and more advanced simulation-based studies,

bridging high-fidelity analysis with practical naval architectural workflows.

Funding

This research received no external funding.

Conflict of interest

The author declares no competing financial or non-financial interests related to this work.

Author contributions

Conceptualization, Methodology, Software, Validation, Formal analysis, Investigation, Data curation, Writing original draft, Writing review & editing: Utku Cem Karabulut.

Data availability

The data that support the findings of this study are available from the author upon reasonable request.

References

- Allan, R. G., Molyneux, D., & Birmingham, R. (2004). Escort tug design alternatives and a comparison of their hydrodynamic performance. *Transactions-Society of Naval Architects and Marine Engineers*.
- Aydın, Ç., Ünal, U. O., Karabulut, U. C., & Sarıöz, K. (2018). Practical computational procedures for predicting steering and braking forces of escort tugs. *Ocean Engineering*, 166, 159–171.
- Bernitsas, M. M., & Kekridis, N. S. (1985). Simulation and stability of ship towing. *International Shipbuilding Progress*, 32(369), 112–123.
- Brandner, P. A. (1995). Performance and effectiveness of omnidirectional stern drive tugs (Doctoral dissertation, University of Tasmania).
- Carrica, P. M., Wilson, R. V., Noack, R. W., & Stern, F. (2007). Ship motions using single-phase level set with dynamic overset grids. *Computers & fluids*, 36(9), 1415–1433.
- CD-ADAPCO, S. (2017). STAR CCM+ user guide version 12.04. CD-Adapco: New York, NY, USA, 62.
- Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H. W., and Raad, P. E. (2008). Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *ASME J. Fluids Eng.*, 130(7), p. 078001.
- Chen, S., Wang, F., Wei, X., Tan, Z., & Wang, H. (2020). Analysis of tugboat activities using AIS data for the Tianjin port. *Transportation Research Record*, 2674(5), 498–509.
- De Luca, F., Mancini, S., Miranda, S., & Pensa, C. (2016). An extended verification and validation study of CFD simulations for planing hulls. *Journal of Ship Research*, 60(02), 101–118.
- Ferziger, J. H., Perić, M., & Street, R. L. (2019). *Computational methods for fluid dynamics*. Springer.

- Fitriadhy, A., & Yasukawa, H. (2011). Course stability of a ship towing system. *Ship Technology Research*, 58(1), 4–23.
- Geerts, S., Vantorre, M., Elout, K., Huijsmans, R., & Fierens, N. (2011). Interaction forces in tug operations. In *2nd International conference on Ship Manoeuvring in Shallow and Confined Water: Ship to ship interaction* (pp. 153–163). Royal Institution of Naval Architects.
- Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of computational physics*, 39(1), 201–225.
- Holtrop, J., & Mennen, G. G. J. (1982). An approximate power prediction method. *International shipbuilding progress*, 29(335), 166–170.
- Hosseini, A., Tavakoli, S., Dashtimanesh, A., Mikkola, T., & Hirdaris, S. (2024). Drift test analysis of a conventional planing hull utilising CFD and 2D+ t models. *Ocean Engineering*, 308, 118226.
- ITTC (2011). Recommended procedures and guidelines: practical guidelines for ship CFD, *26th Int. Towing Tank Conf. 2011*.
- ITTC (2024). Recommended procedures and guidelines: Uncertainty analysis in CFD verification and validation, methodology and procedures, *30th Int. Towing Tank Conf. 2024*, (Report No. 75-03-01-01).
- Karabulut, U. C., & Barlas, B. (2025). A novel approach to full-scale ship resistance predictions: experimental and numerical study of a high-speed planing hull. *Journal of Marine Science and Technology*, 1–25.
- Karabulut, U. C., Özdemir, Y. H., & Barlas, B. (2022). Numerical investigation of the effect of surface roughness on the viscous resistance components of surface ships. *Journal of Marine Science and Application*, 21(3), 71–82.
- Khan, M. M., Das, S., & Karim, M. M. (2025). Numerical investigation of validation and verification for the resistance of JBC hull. *Journal of Maritime Research*, 22(2), 80–86.
- Lutfi, M. A., Prabowo, A. R., Muttaqie, T., Baek, S. J., Do, Q. T., & Cho, J. H. (2023, September). Performance of the designed patrol boat hull: resistance prediction using savitsky method. In *International Conference and Exhibition on Sustainable Energy and Advanced Materials* (pp. 593–598). Singapore: Springer Nature Singapore.
- Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, 32(8), 1598–1605.
- Molyneux, W. D. (2006). Escort tug hydrodynamics: analysis of flow around ships at large yaw angles (Doctoral dissertation, Memorial University of Newfoundland).
- Muzaferija, S. (1998). Computation of free surface flows using interface-tracking and interface-capturing methods. In *Nonlinear water-wave interaction*. Computational Mechanics, Southampton.
- Niklas, K., & Prusko, H. (2019). Full-scale CFD simulations for the determination of ship resistance as a rational, alternative method to towing tank experiments. *Ocean Engineering*, 190, 106435.
- Ozdemir, Y. H., Cosgun, T., & Barlas, B. (2021). Comparative study on ship wave making resistance via viscous and IBEM solvers. *Ocean Engineering*, 241, 110074.
- Patankar, S. V., & Spalding, D. B. (1983). A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. In *Numerical prediction of flow, heat transfer, turbulence and combustion* (pp. 54–73). Pergamon.
- Sarker, D. K., & Tarafder, M. S. (2024). Numerical analysis of fluid flow around ship hulls using STAR-CCM+ with verification results. *Journal of Marine Science and Application*, 23(2), 276–291.
- Savitsky, D. (1964). Hydrodynamic design of planing hulls. *Marine Technology and SNAME News*, 1(04), 71–95.
- Smoker, B., Stockdill, B., & Oshkai, P. (2016). Escort tug performance prediction using computational fluid dynamics. *Journal of Ship Research*, 60(02), 61–77.
- Stern, F., Wilson, R. V., Coleman, H. W., & Paterson, E. G. (2001). Comprehensive approach to verification and validation of CFD simulations—part 1: methodology and procedures. *J. Fluids Eng.*, 123(4), 793–802.
- Weymouth, G. D., Wilson, R. V., & Stern, F. (2005). RANS computational fluid dynamics predictions of pitch and heave ship motions in head seas. *Journal of ship research*, 49(02), 80–97.
- Wilson, R. V., Stern, F., Coleman, H. W., & Paterson, E. G. (2001). Comprehensive approach to verification and validation of CFD simulations—Part 2: Application for RANS simulation of a cargo/container ship. *J. Fluids Eng.*, 123(4), 803–810.
- Xing, T., & Stern, F. (2010). Factors of safety for Richardson extrapolation, IIHR Technical Report No. 469.
- Yusfianda, A. R., Prabowo, A. R., Muhayat, N., Adiputra, R., Smaradhana, D. F., & Nubli, H. (2023, September). Effects of appendage addition on the hull resistance during patrol boat operations. In *International Conference and Exhibition on Sustainable Energy and Advanced Materials* (pp. 603–607). Singapore: Springer Nature Singapore.
- Zhang, S., Wu, Q., Liu, J., He, Y., & Yang, Z. (2022, June). Numerical investigation on hydrodynamic performance of a harbor tug. In *ISOPE International Ocean and Polar Engineering Conference* (pp. ISOPE-I). ISOPE.
- Zhen, L., Wang, K., Wang, S., & Qu, X. (2018). Tug scheduling for hinterland barge transport: A branch-and-price approach. *European Journal of Operational Research*, 265(1), 119–132.
- Zheng, Y., Yiew, L., Jin, Y., Ravindra, K., & Magee, A. R. (2023). Experimental and numerical study on seakeeping performance of a benchmark ASD tug in head seas. *Ocean Engineering*, 272, 113800.